Computational Fluid Dynamics of Facepiece Leakage

Holiday Inn Select, Pittsburgh South
Pittsburgh, PA
John G. Kovac, Physical Scientist
July 19, 2005

Background

- Objectives
  - To develop a computational fluid dynamics (CFD) simulation of the outward leakage of oxygen around the facepiece of a closed circuit breathing device.
  - To experimentally validate the simulation

- Partners
  - NIST Buildings and Fire Research Laboratory

- Timeline
  - Completed before start of FY06
Computational Fluid Dynamics (CFD)

- Computational fluid dynamics (CFD) is the use of computers to analyze problems in fluid flow.
- CFD is a means of visualizing and providing enhanced understanding of the resulting solution.
- Modeling and simulations mean nothing, of course, independent of the reality they are supposed to represent. The accuracy of the simulation must be checked, or validated, against experiment.

Protocol

- Actual heads and masks will be scanned into a 3D data set for entry into the CFD software, providing a physical boundary for the problem to be solved.
- Leak geometries representing an imperfect seal will be defined.
- Oxygen concentration fields and flow streamlines will be computed for multiple leak geometries and for both normal and high stress breathing patterns.
- Model results will be compared to planned experimental work.
Geometrical Modeling

Quality Partnerships Enhance Worker Safety & Health

Thank you!